All output is in the basic coordinate system. Cauchy stresses σ_x (radial), pressure σ_y (axial), σ_z (circumferential), τ_{xy}, pressure $p = 1/3(\sigma_x + \sigma_y + \sigma_z)$, logarithmic strains and volumetric strain are output at the Gauss points.

Pressure loads, specified on the PLOAD4 and PLOADX1 Bulk Data entries respectively, with follower force characteristics, are available for the solid and axisymmetric elements. A pressure load may not be specified on the plane strain elements.

Temperature loads may be specified for all hyperelastic elements on the TEMP and TEMD entries. The hyperelastic material, however, may not be temperature dependent. Temperature affects the stress-strain relation as indicated in Hyperelastic Material, 550.

GPSTRESS and FORCE (or ELFORCE) output is not available for hyperelastic elements.

Bushing Elements

The bushing (generalized spring and damper) elements consist of the following:

- CBUSH
- CBUSH1D

Bush Element (CBUSH)

The generalized spring-damper element CBUSH is a structural scalar element connecting two noncoincident grid points, or two coincident grid points, or one grid point with an associated PBUSH entry. This combination is valid for any structural solution sequence. To make frequency dependent the PBUSH need only have an associated PBUSHT Bulk Data entry. The PBUSHT entry, for frequency dependency, is only used in SOL 108 and SOL 111. The PBUSHT entry can be used to define load-displacement dependency in SOL 106 and SOL 400.

In modal frequency response, the basis vectors (system modes) $[\phi]$ will be computed only once in the analysis and will be based on nominal values of the scalar frequency dependent springs. In general, any change in their stiffness due to frequency will have little impact on the overall contribution to the structural modes.

The stiffness matrix for the CBUSH element takes the diagonal form in the element system:

$$K_o = \begin{bmatrix} k_u & & & & & \\ & k_v & & & & \\ & & k_w & & & \\ & & & k_{\theta_x} & & \\ & & & & k_{\theta_y} & \\ & & & & & k_{\theta_z} \end{bmatrix}$$

For the B matrix replace the k terms with b.

When transformed into the basic system, there is coupling between translations and rotations, thus ensuring rigid body requirements.

The element axes are defined by one of the following procedures:

1. If a CID is specified then the element x-axis is along T1, the element y-axis is along T2, and the element z-axis is along T3 of the CID coordinate system. The options GO or (X1,X2,X3) have no meaning and will be ignored. Then $[T_{ab}]$ is computed directly from CID.
2. For noncoincident grids ($\text{GA} \neq \text{GB}$), if neither options GO or (X1,X2,X3) is specified and no CID is specified, then the line AB is the element x-axis. No y-axis or z-axis need be specified. This option is only valid when K1 (or B1) or K4 (or B4) or both on the PBUSH entry are specified but K2, K3, K5, K6 (or B2, B3, B5, B6) are not specified. If K2, K3, K5, K6 (or B2, B3, B5, B6) are specified, a fatal message will be issued. Then $[T_{ab}]$ is computed from the given vectors like the beam element.

Direct Frequency Response

Nominal Values

The following matrices are formed only once in the analysis and are based on the parameter input to EMG of /' '/ implying that the nominal values only are to be used for frequency dependent springs and dampers.

$$K_{dd} = (1 + ig)K^1_{dd} + K^{2x}_{dd} + iK^4_{dd}$$

$$M_{dd} = M^1_{dd} + M^2_{dd}$$

$$B_{dd} = B^1_{dd} + B^{2x}_{dd}$$

ESTF

The following matrices are formed at each frequency in the analysis and are based on the parameter input to EMG of /'ESTF'/

$$\Delta K^f_{dd} = (1 + ig)\Delta K^{1f}_{dd}$$

$$\Delta K^{4f}_{dd} = g^f_e[\Delta k^f_j] \quad j = 1 \rightarrow 6$$

$$\Delta B^f_{dd} = \Delta B^{1f}_{dd}$$

ESTNF

The following matrices are formed at each frequency in the analysis and are based on the parameter input to EMG of /'ESTNF'/

$$\Delta K^{4f\,nominal}_{dd} = [\Delta g^f_e\, k^{f\,nominal}_j] \quad j = 1 \rightarrow 6$$

and j runs through the stiffness values specified for the CBUSH element or j = 1 for the CELAS1 and CELAS3 elements.

Then at each frequency form:

$$K_{dd} \leftarrow K_{dd} + \Delta K^f_{dd} + i\Delta K^{4f}_{dd} + i\Delta K^{4f\,nominal}_{dd}$$

$$B_{dd} \leftarrow B_{dd} + \Delta B^{f}_{dd}$$

Then the equation to be solved is:

$$[-\omega^2 \ M_{dd} + i\omega \ B_{dd} + K_{dd}][u_d] = [P_d]$$

Modal Frequency Response

Basis Vector and Nominal Values

The basis vector matrix $[\phi]$ (system modes) is formed only once in the analysis using nominal values for frequency dependent elements.

The following matrices are formed only once in the analysis and are based on the parameter input to EMG of /' '/ implying that the nominal values only are to be used for frequency dependent springs and dampers.

$$K^2_{dd} = K^{2x}_{dd} + igK^1_{dd} + iK^4_{dd}$$

$$B^2_{dd} = B^1_{dd} + B^{2x}_{dd}$$

ESTF

The following matrices are formed at each frequency in the analysis and are based on the parameter input to EMG of /'ESTF'/

$$\Delta K^{2f}_{dd} = ig\Delta K^{1f}_{dd}$$

$$\Delta K^{4f}_{dd} = g^f_e[\Delta k^f_j] \quad j = 1 \to 6$$

$$\Delta B^{2f}_{dd} = \Delta B^{1f}_{dd}$$

ESTNF

The following matrices are formed at each frequency in the analysis and are based on the parameter input to EMG of /'ESTNF'/

$$\Delta K^{4f\ nominal}_{dd} = [\Delta g^f_e \ k^{f\ nominal}_j] \quad j = 1 \to 6$$

and j runs through the stiffness values specified for the CBUSH element or j = 1 for the CELAS1 and CELAS3 elements.

Then at each frequency form:

$$K^2_{dd} \leftarrow K^2_{dd} + \Delta K^f_{dd} + i\Delta K^{4f}_{dd} + i\Delta K^{4f\ nominal}_{dd}$$

$$B^2_{dd} \leftarrow B^2_{dd} + \Delta B^f_{dd}$$

The GKAM module will then produce:

$$K_{hh} = [k] + [\Phi]^T[K^2_{dd}][\Phi]$$

$$B_{hh} = [b] + [\Phi]^T [B^2_{dd}][\Phi]$$

$$M_{hh} = [m] + [\Phi]^T [M^2_{dd}][\Phi]$$

Then the equation to be solved is:

$$[-\omega^2 M_{hh} + i\omega B_{hh} + K_{hh}][u_h] = [P_h]$$

Element force calculation:

Frequency:

$$F_e = [(1 + ig_t)K_e + i\omega B_e]U_e$$

where

$$g_t = g + g_e$$

Static:

$$F_e = K_e U_e$$

Transient:

$$F_e = K_e U_e(t) + \left[B_e + \left(\frac{g}{w3} + \frac{g_e}{w4}\right)K_e\right]\dot{U}_e(t)$$

CBUSH Stiffness and Structural Damping Matrices

If both stiffness K and structural damping ge is specified each with their own frequency dependent tables, matrix terms of the following form are created:

$$K_j^f + i \cdot ge_j^f \cdot K_j^f$$

If stiffness K with frequency dependent tables and non frequency dependent structural damping ge are specified, matrix terms of the following form are created:

$$K_j^f + i \cdot ge_j^0 \cdot K_j^f$$

If non frequency dependent stiffness K with frequency dependent structural damping ge are specified, matrix terms of the following form are created:

$$K_j^0 + i \cdot ge_j^f \cdot K_j^0$$

In the above three expressions, the superscript f denotes frequency dependent and the superscript 0 denotes a nominal value. The subscript j implies the j-th degree of freedom of the CBUSH element. The real term goes into the element stiffness matrix and the imaginary term goes into the element K^4 matrix.

Bush1D Element (CBUSH1D)

The BUSH1D is a one dimensional version of the BUSH element (without the rigid offsets) and supports large displacements.

The element is defined with the CBUSH1D and a PBUSH1D entry. The user may define several spring or damping values on the PBUSH1D property entry. It is assumed that springs and dampers work in parallel. The element force is the sum of all springs and dampers.

The BUSH1D element has axial stiffness and axial damping. The element includes the effects of large deformation. The elastic forces and the damping forces follow the deformation of the element axis if there is no element coordinate system defined. The forces stay fixed in the x-direction of the element coordinate system if the user defines such a system. Arbitrary nonlinear force-displacement and force-velocity functions are defined with tables and equations. A special input format is provided to model shock absorbers.

Benefits

The element damping follows large deformation. Arbitrary force deflection functions can be modeled conveniently. When the same components of two grid points must be connected, we recommend using force-deflection functions with the BUSH1D element instead of using NOLINi entries. The BUSH1D element produces tangent stiffness and tangent damping matrices, whereas the NOLINi entries do not produce tangent matrices. Therefore, BUSH1D elements are expected to converge better than NOLINi forces.

Output

The BUSH1D element puts out axial force, relative axial displacement and relative axial velocity. It also puts out stress and strain if stress and strain coefficients are defined. All element related output (forces, displacements, stresses) is requested with the STRESS Case Control command.

Guidelines

The element is available in all solution sequences. In static and normal modes solution sequences, the damping is ignored.

In linear dynamic solution sequences, the linear stiffness and damping is used. In linear dynamic solution sequences, the BUSH1D damping forces are not included in the element force output.

In nonlinear solution sequences, the linear stiffness and damping is used for the initial tangent stiffness and damping. When nonlinear force functions are defined and the stiffness needs to be updated, the tangents of the force-displacement and force-velocity curves are used for stiffness and damping. The BUSH1D element is considered to be nonlinear if a nonlinear force function is defined or if large deformation is turned on (PARAM,LGDISP,1). For a nonlinear BUSH1D element, the element force output is the sum of the elastic forces and the damping forces. The element is considered to be a linear element if only a linear stiffness and a linear damping are defined and large deformation is turned off.

Limitations

1. The BUSH1D element nonlinear forces are defined with table look ups and equations. Equations are only available if the default option ADAPT on the TSTEPNL entry is used, equations are not available for the options AUTO and TSTEP.
2. The table look ups are all single precision. In nonlinear, round-off errors may accumulate due to single precision table look ups.
3. For linear dynamic solution sequences, the damping forces are not included in the element force output.
4. The “LOG” option on the TABLED1 is not supported with the BUSH1D.

Interface Elements

The interface elements are primarily used when performing global local analyses. The current implementation is for p-elements.

For curve interfaces the following entries are used:

- GMBNDC -- Geometric Boundary - Curve
- GMINTC -- Geometric Interface - Curve
- PINTC -- Properties of Geometric Interface - Curve

For surface interfaces the following entries are used:

- GMBNDS -- Geometric Boundary - Surface
- GMINTS -- Geometric Interface - Surface
- PINTS -- Properties of Geometric Interface - Surface

The interface elements use a hybrid variational formulation with Lagrange multipliers, developed by NASA Langley Research Center. There are displacement variables defined on the interface element, in order to avoid making the interface too stiff, such as a rigid element. There are also Lagrange multipliers defined on each boundary, which represent the forces between the boundaries and the interface element. This formulation is energy-based and results in a compliant interface.

Curve Interface Elements

The addition of interface elements allows dissimilar meshes to be connected over a common geometric boundary, instead of using transition meshes or constraint conditions. Primary applications where the analyst specifies the interface elements manually include: facilitating global-local analysis, where a patch of elements may be removed from the global model and replaced by a denser patch for a local detail, without having to transition to the surrounding area; and connecting meshes built by different engineering organizations, such as a wing to the fuselage of an airplane.

Primary applications where the interface elements could be generated automatically are related to automeshers, which may be required to transition between large and small elements between mesh regions; and h-refinement, where subdivided elements may be adjacent to undivided elements without a transition area.